

ECE322L - Lab 1

Circuit Simulation

Goal

Explore and gain understanding of Pspice and Multisim software.

Software needed

- ❑ PSpice and MultiSim

The Lab

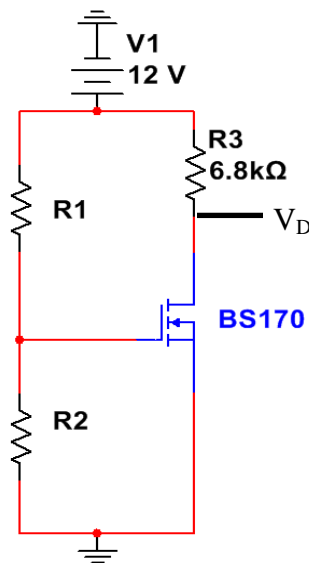


Figure 1: Circuit Diagram.

Part 1 – Hand Calculations

Let $K'_n = 1.825 \text{ mA/V}^2$, $(W/L) = 1$, $V_{TN} = 1 \text{ V}$, and $R1 + R2 = 100\text{K}$. Bias the transistor such that half the supply voltage is across the transistor.

1. Find I_{DQ} such that half the supply voltage is across the transistor
2. Find $R1$ and $R2$ through hand calculations.
3. Solve for V_{DSQ} and V_{GSQ} . Use the following table to keep track of the measurements.

	Hand Calc	Pspice	Multisim
I_{DQ}			
V_{DSQ}			
V_{GSQ}			
R1		N/A	
R2		N/A	

Part 2 – Modeling the Circuit in Pspice

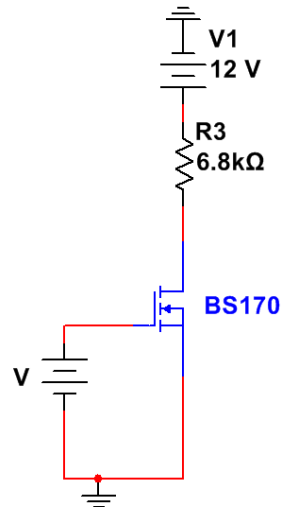


Fig 2: Pspice circuit diagram

1. Create the circuit in Pspice

First line is always a comment. Comments denoted by ‘’

*Supply voltage, connected to node 1 and 0, 0 is always defaulted to ground

V1 1 0 12

*Gate Voltage, we will use this to sweep for V_{GSQ}

Vg 2 0 5

*Drain Resistor < + node> < - node > <Resistance>

Rd 1 3 6.8k

*<label> <drain> <gate> <source> <bulk> <.model name>

M1 3 2 0 0 ntype

.model ntype nmos vto=1 kp=1.825m l = 1.0u w = 1.0u

.probe

*operating point

.op

.end

*.model <model name> <type (NMOS, PMOS, NJF, PJF, NPN, PNP)> <parameters>

*here we have set threshold voltage (vto) equal to 1v, and $K' = 1.825\text{m}$

2. In order to find out what gate voltage is needed to provide half the supply voltage across the transistor we can use the DC sweep command:

.dc <voltage or current node> <starting point> <end point> <step size>

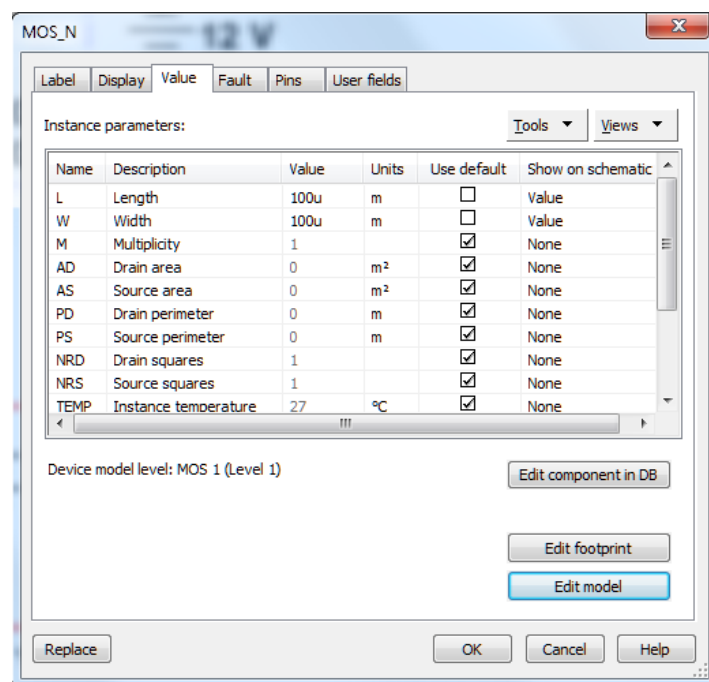
Place this command above the .probe statement

3. Provide a graph with V_D as a function of V_G in your report.
4. Set V_g to the voltage that provides half the supply across V_{DS}
5. Include the output file and the quiescent values in your report.

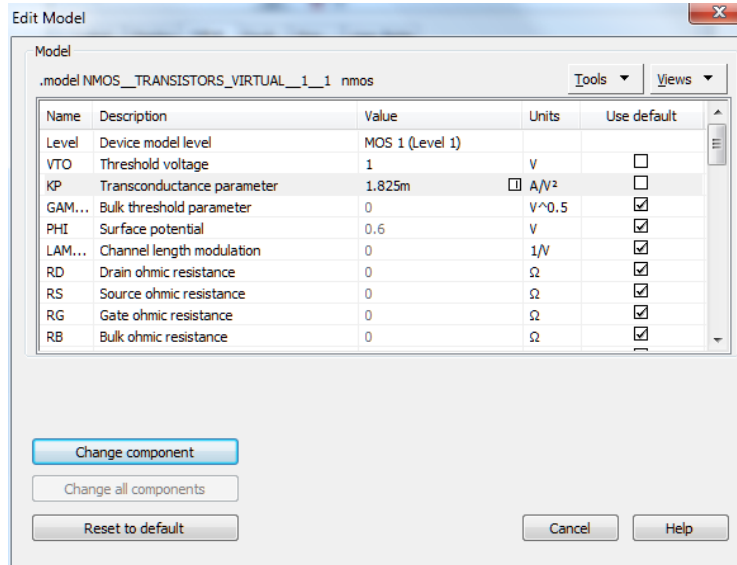
Note: You may have to save the file as a .cir and then reopen the file in order to run simulations.


Part 3 - Multisim

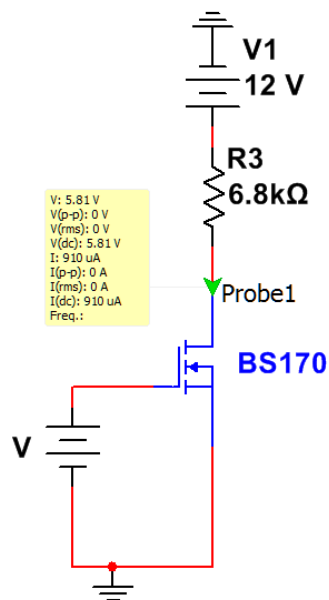
1. Create the circuit shown in figure 2.
 - a. You can add components by right clicking the design area and selecting 'Place component', selecting 'Place -> Component' from the top menu, or by entering 'Ctrl-w'
 - b. You can add junctions on wires to aid in wire connections by entering 'Ctrl-J' and then clicking where you'd like the junction added.
 - c. Use the MOS_N transistor, located in TRANSISTORS_VIRTUAL
 - i. After placing the MOS_N transistor we must edit the .model. Do this by double clicking the transistor in the schematic. Then click on "Edit model" near the bottom right.



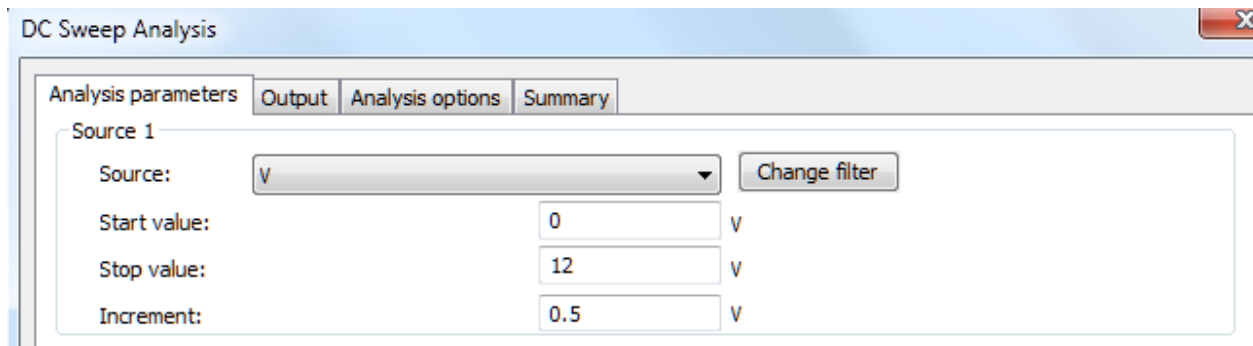
- ii. Enter the given VTO and KP values, then click change component.



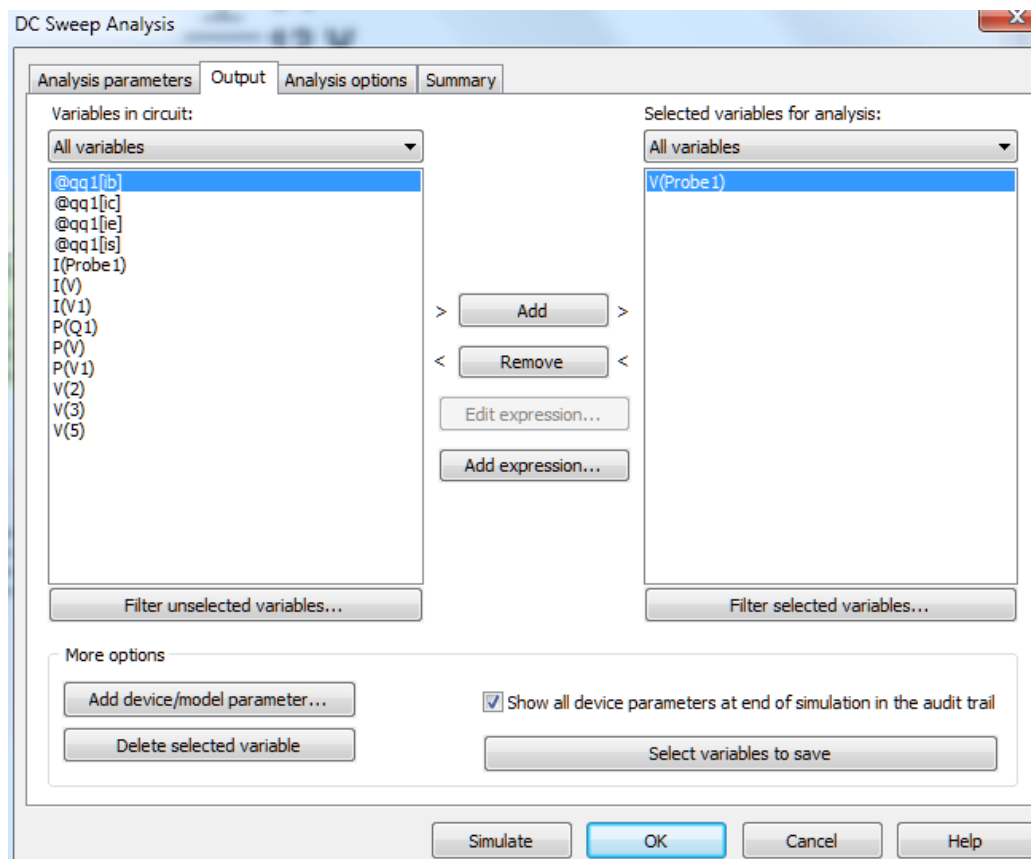
2. In Multisim we are able to place probes which can help with quick analysis and setting up simulations. You can place these probes using the  button located near the bottom of the toolbar on the right side of the window. Click the button then click where you'd like the probe to be.



3. Now let's find V_G needed using multisim:
 - a. From the menu toolbar select Simulate -> Analysis -> DC Sweep
 - i. From here we need to set up the DC Sweep:



- ii. We also need to select what output we are interested in, then click simulate:



4. Provide a screen capture of the DC sweep graph and include Multisim quiescent values on your table.
5. Compare the differences of the quiescent values obtained through Pspice and hand calculations. What could cause the differences?
6. Which simulation software do you prefer? Why? (not limited to Pspice and Multisim)

Rubric for your report:

Lab 1 Rubric	
Requirement:	Possible Points
Hand Calculations: a. R1 and R2 (10 points) b. Quiescent Values (15 points)	25
Pspice a. Quiescent Values with output file (15 points) b. Graph of V_D as a function of V_G (10 points)	25
Multisim a. Quiescent Values (15 points) b. Graph of DC sweep (10 points)	25
Comparison (15 points)	15
Simulation program preference (10 points)	10